

\$250

QTC

November 2024



The Radio Amateur Society
of Australia

WIA Loses Monopoly on Repeaters, Beacons
VKFF National Get Together
Design Your Own Circuit Boards

**RANDOM
DX CONTACT**

**Clubs - Advertise Courses, Exams in the
Foundation Level Study Guide**

**CAUSING
QRM**

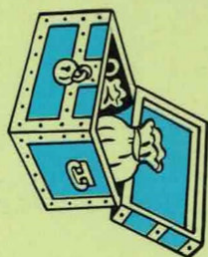
**SHUT DOWN
UNTIL
PROBLEM
RESOLVED**

IC7300



\$1,700

**HAMFEST
SALE**

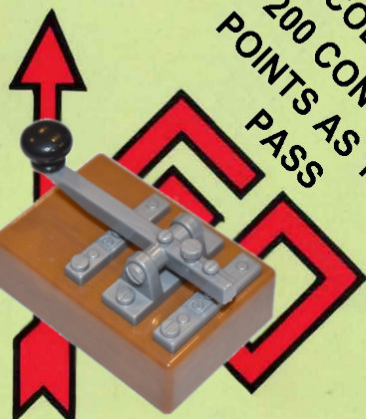


FTDX101D



\$5,000

**COLLECT
200 CONTEST
POINTS AS YOU
PASS**



**ANTENNAPALOOZA
FREE**



**REPEATER
APPLICATION
REQUIRES
WIA
APPROVAL
(Wait 6 MONTHS)**

**PERTH TECH
FREE**



CHAN



In This Issue:

From the Editor	3
From the President	4
WIA Loses Monopoly on Repeater and Beacon Application Process	5
VKFF National Get-Together	6
Clubs - Advertise Your Courses in the Foundation Study Guide	8
A Visitor's Take on PerthTech	9
Foundation Level Study Guide.	11
Supporting RASA	12
Designing Your Own Circuit Boards	13

Publication Information

Contact: info@vkradioamateurs.org
Website: <http://vkradioamateurs.org>

QTC is a publication of The Radio Amateur Society of Australia Inc.

ABN: 64 687 227 446 (RASA)

QTC is published monthly. If you would like to receive your copy you can either visit our web site to download or send us an email and we'll put you on our distribution list.

You are welcome to join RASA or simply make a donation to support our work.

QTC is Copyright © 2024

Email us at info@vkradioamateurs.org

Cover picture credit:
Ian Jackson VK3BUF

Contributing Items For QTC Magazine

QTC Magazine welcomes contributions for future editions. When planning to submit an article, please read our submission guidelines first.

Following the guidelines will save you and the editing team a lot of time and effort. The guidelines are [HERE](#)



New to the hobby? The RASA Welcome to Amateur Radio Guidebook provides an introduction to our hobby for new Foundation class amateurs.

The book is published digitally. It contains many hotlinks to external websites with useful information.

It is available [HERE](#) as an Acrobat pdf file suitable for reading on a PC or tablet. It can be printed if required.



From the Editor



With the introduction of the Amateur Radio Class Licence in Australia early this year, Amateurs expressed concern that there was no “licence document” issued for Amateurs who held a licence prior to the changeover.

Newly qualified Amateurs will be issued with a Certificate of Recognition on passing the relevant examination, however Amateurs already licenced at the time were issued with a letter authorising them to operate under the class licence.

We’ve heard stories concerning external administrations or individuals providing services to Amateurs that require some form of registration, denying the service to Australian Amateurs because they could not produce a current “licence” document.

The matter has been raised with the ACMA, asking them to reconsider, providing a certificate on request, however their response is a firm “no”.

The ACMA further say that where instances of refusal to accept the Australian Letter of Authority have been referred to them, every case has been resolved satisfactorily.

Whether or not we like it, the ACMA remains firm in its view that the letter is sufficient, and that where its validity has been questioned, the matter has been resolved when referred to the ACMA. The Letter of Authority provides all the required information.

We must accept, for the time being at least, that we won’t have a nice certificate for our shack wall, and that perhaps our energy for now should be directed to other outstanding issues such

as access to 5 MHz and one kilowatt for Advanced licensees.

In other matters concerning the ACMA, that body recently released its new policy and process for obtaining an Amateur Repeater or Beacon licence.

It’s both pleasing and rewarding to see that the ACMA accepted several key recommendations made by RASA in our response to the consultation on the Repeater and Beacon application process earlier this year.

See the full report on page 5.

Elsewhere in this edition of QTC, there’s a report on the recent VKFF National Gathering in Bendigo. Congratulations to Paul and Marija for organising another successful amateur radio event.

Enjoy your November read of QTC



Check out [QRm.GURU](https://qrm.guru) to learn how to track down and reduce QRM, either by following the described process or reading the examples already there.

From the President

The cost of nostalgia

I dusted off my Yaesu FT7 Transceiver, an entry level radio way back in 1978. Back then I worked many countries, mostly portable and enjoyed the thrill of accomplishing this using a simple setup. Today, it looks so basic and unlike fine



I quite like the G90 and have witnessed its capabilities, especially when it comes to activating parks and other portable "On the Air" activities. The G90 offers great value, especially when you compare it to the 2024 adjusted price of the FT7.

Today Amateur Radio is much cheaper than it was. Some things are even free!*

Last month RASA revised two of their training and reference tools.

The second edition of the **Foundation Level Study Guide** is now available, featuring several updates and improvements. This study guide includes chapters filled with text, videos, and external resources to enhance your learning experience, making it a valuable reference tool. And the best part? It's completely free! Go to our [website](#) to access the current version.

The other is **The Australian Amateur Radio Regulations Handbook**, a plain English guide to Amateur Radio regulations. This handbook stands out as the sole study resource focused on regulations, making it perfect for anyone aiming to achieve either a Standard or Advanced qualification. With the change to Class Licensing, the handbook has been revised accordingly. You can find the current version at <https://vkregs.info/>.

RASA offers a follow-up resource to the Foundation Level Study Guide called the **Welcome to Amateur Radio Guidebook**. You can download the latest updated version [HERE](#).

There are other resources in development.

Not bad for a modern national body who charges \$10 for membership!

The cost of Amateur Radio has never been lower than it is today.

73
Paul VK2APA



wine, it has not improved with age. The receiver has some frustrating intermittent issues, and the transmit output is lower than the original 10 watts. However, I will repair it because it holds a special nostalgic value for me.

The FT7 set me back \$535 on sale at Dick Smiths, which was quite a bit for a trainee technician! Adjusted for inflation, that's about \$3,305.27 today - a staggering sum, especially when you can now get an ICOM IC7300 for less than \$1,500.

Nostalgia sells, as anyone who goes to a club's Trash and Treasure can attest. One man's treasure is another man's recycling, and ancient equipment full of trouble and obsolescence sell for many \$\$\$\$. Unless you enjoy dedicating a lot of time to diagnosing issues, hunting for parts, and you have the expertise, I suggest you consider buying new equipment or keep an eye out for great deals on newer used gear that occasionally becomes available.

WIA Loses Monopoly on Repeater and Beacon Application Process

As a part of the transition to a Class Licence, the ACMA committed to update the Amateur Repeater and Beacon Application process.

Unlike operator qualifications which have transitioned to a Class License framework, Repeaters and Beacons remain as Apparatus Licences and continue to require annual licence renewals with applicable fees.

The existing process has been a contentious issue for many years. The ACMA would not approve any repeater application that did not have a letter of endorsement from the WIA.

RASA received complaints detailing examples of unnecessarily long delays, preferential treatment, undocumented specifications, and unauthorised changes to applications by the WIA.

RASA has lobbied the ACMA on these matters on several occasions. RASA's position has consistently been that applicants should be able to apply for a repeater licence without compulsory involvement by the WIA.

It is not the role of a representative body to dictate how and where Repeaters and Beacons should be located.

For too long the WIA operated a process which lacked transparency and accountability.

This year the ACMA conducted a consultation process for a new repeater and beacon assignment process.

The WIA maintained that they should retain exclusive endorsement rights for all Australian repeater and beacon applications.

On 30th October 2024, the ACMA circulated a bulletin detailing the new process that would apply to repeater QTC November 2024

applications. The document is somewhat complex and should be read in full by anyone seeking a new repeater application. Read it [HERE](#).

One of the key takeaways from this new process is that applicants no longer have to seek approval specifically from the WIA. The new document states:

The licensing arrangements detailed in this FAP remove the previous requirement to obtain a Wireless Institute of Australia (WIA) endorsement letter for standard applications, which represent the bulk of amateur assigned licences. This improves both the efficiency and transparency of the amateur beacon and repeater licensing process.

It should also be noted that the ACMA will accept repeater applications where the applicant has either a Standard or Advanced qualification and not advanced only. This is also a welcome development.

Other references within the assignment process remove direct reference to the WIA and have replaced the wording to more general 'amateur representative bodies'.

For example: *Applicants are strongly encouraged to discuss these proposed applications with the relevant amateur representative bodies prior to contacting the ACMA.*

From RASA's perspective we are pleased to see that the ACMA has adopted a more balanced and impartial approach within these new processes.

The result is that it should now be a faster and easier repeater and beacon application process.





VKFF National Get-Together

On Friday 25th October 2024, radio amateurs from around Australia descended upon the central goldfields city of Bendigo Victoria to attend the VKFF National Get-Together. The event is a gathering of VKFF park enthusiasts for the World Wide Flora Fauna (WWFF) program.

This was the second year that the event had been held. Last year about 35 amateurs attended the Get-Together in Renmark, South Australia.

This year's event commenced with an evening dinner at the historic Hotel Shamrock, a grand hotel in the heart of Bendigo, built in 1864. About 50 people enjoyed a magnificent meal and a story or two about VKFF park activating.

On Saturday 26th, about 50 amateurs attended the presentation segment of the Get-Together which was held at the Quality Hotel Lakeside at Bendigo. Our MC for the day was Chris VK5FR.

Each attendee received a welcome pack and for those who had previously placed orders, they were handed their Get Together merchandise which included polo shorts, caps, beanies, stubby holders, and coffee mugs.

The day commenced with a welcome by myself the VKFF National Co-Ordinator followed by a presentation I delivered on an introduction to the WWFF/VKFF program. Peter VK3ZPF then delivered a valuable presentation on Tips for



QTC November 2024

2024 VKFF NATIONAL GET TOGETHER



Bendigo, Victoria

Activators and Hunters. Hans VK6XN followed with details on mapping and how to find a VKFF park.



We then took a break for morning tea and enjoyed scones, jam and cream, and tea and coffee.

After the morning tea break Peter VK3PWG and Malcolm VK3OAK spoke about their recent trip to King Island where they activated several VKFF parks. This was followed by David VK3TUN who spoke about the use of satellites from parks. Tameeka from Greater Bendigo Snake Control then delivered a very interesting presentation on snake awareness and safety.

After our lunch break, Marija VK5MAZ spoke about the extremely useful parksneaks website. This was followed by Peter VK3PF who spoke about WWFF activator logs and the WWFF Logsearch database.

We then enjoyed afternoon tea

consisting of various Danish, tea and coffee. Our afternoon presenters were Andy VK5LA who spoke about CW and digital modes from VKFF parks. This was followed by Ivan VK5HS who delivered a presentation on antennas and power supplies.



This was followed by a question-and-answer session with some of the VKFF Team members who were in attendance, including myself, Ian VK1DI, Gerard VK2IO, Hans VK6XN, and Mark VK5MK. A number of certificates were then presented to the VKFF Team members and to those who



had delivered presentations during the day.

A raffle was held which raised a total of \$529.00 with the proceeds being donated to Trees for Life. Some excellent prizes were on offer including a solder station donated by JMM Solutions, a UHF CB radio starter kit donated by RivComm Radio Solutions, \$200.00 worth of gift vouchers donated by Jaycar Electronics, a squid pole donated by Haverford, and Get Together



merchandise donated by the VKFF Team.

Joe VK3YSP and Julie VK3FOWL had on display their pedestrian mobile station. Various other equipment was also on display.

On Saturday evening we returned to the Quality Hotel Lakeside and enjoyed an excellent meal in our own private function room. Our guest speaker for the evening was Jordan Crook, Parks and Nature campaigner from the Victorian National Parks Association. Jordan's talk was captivating, speaking on his work of caring for wildlife and natural places.

On Sunday morning a BBQ breakfast was hosted by the Bendigo Amateur Radio & Electronics Club. This was followed by dozens of VKFF park activations around the Bendigo region. The bands were certainly alive with lots of VKFF activity.

I would like to thank all those who attended this year's VKFF National Get Together, and a big thank you to my wife Marija VK5MAZ for helping me organizing this year's event.

The 2025 VKFF National Get Together will be held in Canberra, ACT. More information to follow on the [WWFF Australia website](#).

73 & 44, Paul VK5PAS
VKFF National Co-Ordinator.



ANTENNAPALOOZA Back for 2024

Antennapalooza is returning. This Field weekend is free, sponsored by local clubs and RASA. It will take place on Saturday November 30 and December 1 in Drouin West. Setup will take place on Friday the 29th.

No booking required. Location as before is at **408 Old Sale Road, Drouin West**

This year the theme is Wire Antennas. A demonstration Curtain Array on 20M built for the event should be up and running. This is a large wire antenna supported between two 18 metre posts, 60 metres apart.

Various talks and demo's will be conducted under the (improved) pavilion.

Visit for the day, or camp for the weekend. Set up your own antenna.

View details at the Antennapalooza website at: <https://antennapalooza.net.au/>

Clubs - Advertise Your Courses in the Foundation Study Guide

Last month RASA announced the release of an update to the Foundation Level Study Guide (FLSG). It included new content, including the complete Foundation syllabus and a new advertising section on the last page.

As it is completely free, this study guide is downloaded hundreds of times per month from many prospective Amateurs across the country. If you or your club conduct exams or classes, you can have your contact details listed free of charge at the end of the FLSG.

This free service is provided to the Amateur Radio community by RASA.

Simply email your course, exam and contact details to info@vkradioamateurs.org and it will be added to the FLSG.

RASA
The Radio Amateur Society of Australia

Needs Your Support

RASA PRODUCES HUGE RESULTS FROM A TINY BUDGET

- Foundation Level Study Guide
- QTC Magazine
- QRM Guru
- ARTS, VKRegs
- Representation to Government

Study Guide

QRM.guru
Resolving Interference for Amateur Radio

Welcome to Amateur Radio
A guidebook for newcomers

YF W DX Contest
An unlimited opportunities for you to explore the technical, scientific and social aspects of this truly global hobby

Click Here to Donate or Pay membership

QTC
been developed by RASA and is provided free of charge.

RASA relies on member fees and donations, so that we can offer our resources free to everyone. We're inviting you to sign up or renew your membership, or simply donate. You can do this at our



A Visitor's Take on PerthTech

By Andy Beales VK4KCS.

I needed to travel to Perth W.A. recently, for family reasons, and took the opportunity to make this trip coincide with PerthTech 2024, held at the Gidgegannup Recreation Centre on Saturday 21st of September.

This event stands out from many of the other 'xxxTech' events I have attended, for the wide range of very knowledgeable speakers, covering subjects as diverse as:

Using Machine learning to improve Digital Voice audio quality, by Mooneer Salem K6AQ via Zoom, to:

Chris Chapman VK3QB discussing and receiving plenty of feedback about the project for the harmonisation of the Bandplan for 40 Metres, to:

Dennis and Dave's fascinating discussion about measuring the Permittivity of uncommon Dielectrics.

The program included below demonstrates the wide range of subjects covered.

The Programme for PerthTech 2024, Hosted by WA Amateur Radio News

Mooneer Salem K6AQ - Presentation: Machine Learning and FreeDV

Digital voice (especially on VHF/UHF) is something that hams have been doing for several decades. While it does have advantages over traditional analog voice, one big disadvantage is the lower perceived audio quality. This is despite being able to decode and copy digital voice transmissions at lower signal to noise ratios than would otherwise be possible. HF, meanwhile, adds additional complications that affect the ability to decode a signal that's of acceptable audio quality. He spoke about how the FreeDV project is using machine learning to improve digital voice on the HF bands.

Emeritus Prof. Peter Hall, VK6HP built a LF/MF station

Low and medium-frequency radio often appears arcane and difficult, while remote station operation frequently involves specialized IT solutions. Peter described how new-generation radio and related hardware (both commercial and homebrew) and standard remote desktop software, together with a 4G LTE data link, made possible a world-class LF/MF station at Manjimup, south of Perth. The station is built around a de-commissioned aeronautical NDB facility, the morse ident of which was "MJM".

Peter Miles VK6YSF discussed how necessity propelled him into maintenance of his radio equipment.

He walked us through a couple of example repairs, what he did wrong and what he learned from the experience.

Peter Bassett-Scarfe VK6PBS and Co-presenter Chris Stickland, ESM, Manager. State Emergency Service Unit spoke on "Amateur Radio Emergency Service Support"

The Mandurah State Emergency Service (unit) and the Peel Amateur Radio Group Inc have had a close relationship since 1983.

QTC November 2024

Page 9



Some amateur radio operators were also volunteers of the Mandurah SES at the time PARG was formed. Today some members of PARG are also members of the Mandurah SES.

PARG has a formal Memorandum of Understanding (MoU) with Mandurah SES. PARG has a specific callsign for use on amateur radio frequencies for training exercises and operations with Mandurah SES. It is VK6SES.

Mandurah SES has allocated a callsign to the PARG Mobile Communications Trailer for use on SES channels for training and operations involving PARG.

There is also an arrangement termed PARGESS - the Peel Amateur Radio Group Emergency Service Support. "PARG maintains a supportive stance to assist the radio communications needs of the emergency services in and around Mandurah."

Bob Lockley VK6KW described how WICEN in Western Australia is currently undergoing a revival.

Chris Chapman VK3QB spoke about the 40 metre bandplan review being undertaken by the International Amateur Radio Union (Region 3) iThis is being done under the banner of Harmonisation, and seeks to address many of the inconsistencies in the 40 metre band plans across different countries and regions. How will this new plan affect you, especially if you are a 40 metre CW operator? Chris led a discussion.

Dennis Brown VK6AKR and Dave VK6KV spoke about "Measuring Permittivity of Uncommon Dielectrics" with the sub-title of "Spend more time on air, less time tuning that new antenna"

Randall Wayth VK6WR made a presentation about "Image and Data Transmission over the air with Digital SSTV"

The Venue

The venue where the event was held is big enough for the attendees to be comfortably seated in a half circle rows around the speaker without being crowded together but not too far away for easy listening and conversation for the inevitable Q & A sessions with the Speaker.

An excellent Lunch is included in the very moderate Ticket price, Tea and Coffee are continuously available.

Any Amateur Radio operators can expand their practical knowledge about the ins and outs of this great hobby by taking on board the kind of research and results that is presented at these get togethers.

Keep an eye on the vk6.net website for the date for PerthTech 2025. It is bound to be as valuable as I found this one.

73 Andy VK4KCS

Editor's Note: Andy is a RASA Board Member

The Radio Amateur Society of Australia



Foundation Level Study Guide

V2 October 2024



Your Gateway to the World of Amateur Radio

Amateur Radio provides virtually unlimited opportunities for you to explore the technical, scientific and social aspects of this truly global hobby.

A Free Amateur Radio Learning Resource

This multimedia learning resource has been developed by RASA and is provided free of charge.



info@vkradioamateurs.org

[Web: vkradioamateurs.org](http://Web:vkradioamateurs.org)

Foundation Level Study Guide.

RASA developed this resource to make your entry into the hobby easier and cheaper.

Version 2 was recently released, so if you haven't updated, you can do it now.

Like all our resources, the FLSG is free, and we encourage you to share it with your club, friends and others who may be interested in becoming a radio amateur.

The resource is backed up by our other award-winning resources, including QRM Guru, VKRegs and the Amateur Radio Tech Support knowledge system and helpdesk.

You can download the Foundation Level Study Guide [HERE](#).



- 2 cups of coffee,
- or
- 6.5 litres of petrol,
- or
- 4 or 5 days of good Internet access,
- or
- 4 bottles from a six pack of beer,
- or
- 1 kg of cheap BBQ sausages,
- or
- A small fast food meal deal,
- or
- 1 months base subscription to an internet streaming service,
- or
- 83 minutes of street parking in the Sydney CBD,
- or
- 260 \$10 dollar notes will buy you an ICOM IC-7300,
- or
- 40.5 days membership to another Amateur Organisation,
- or.....

A year's RASA membership, which allows us to continue the work of...

- **Active two way engagement and representation of all Australian Radio Amateurs by a National Body to the regulator.**
- **A free magazine with member regular bulletins.**
- **Free RASA Resources**
- **Foundation License Study Guide** Dynamic and interactive training.
- **VKREGS.INFO V2** Regulations in simple English.
- **Welcome to Amateur Radio V2** Where to go next once licenced.
- **VK3RASA Reverse Beacon**, part of the Reverse Beacon Network Propagation tool.
- Sponsorship of **QRM GURU**, an internationally recognised tool for the management of interference.
- RASA is exploring **strategies to enhance and expand Amateur Radio**, as previous incentives are no longer effective in the current landscape. This is a significant initiative, as the hobby must evolve to remain relevant. Stay tuned to QTC Magazine for updates on our progress.
- And other services.



**BEING HEARD
IS IMPORTANT**

The Radio Amateur Society of Australia inc.
vkradioamateurs.org

Designing Your Own Circuit Boards

Ian Jackson VK3BUF

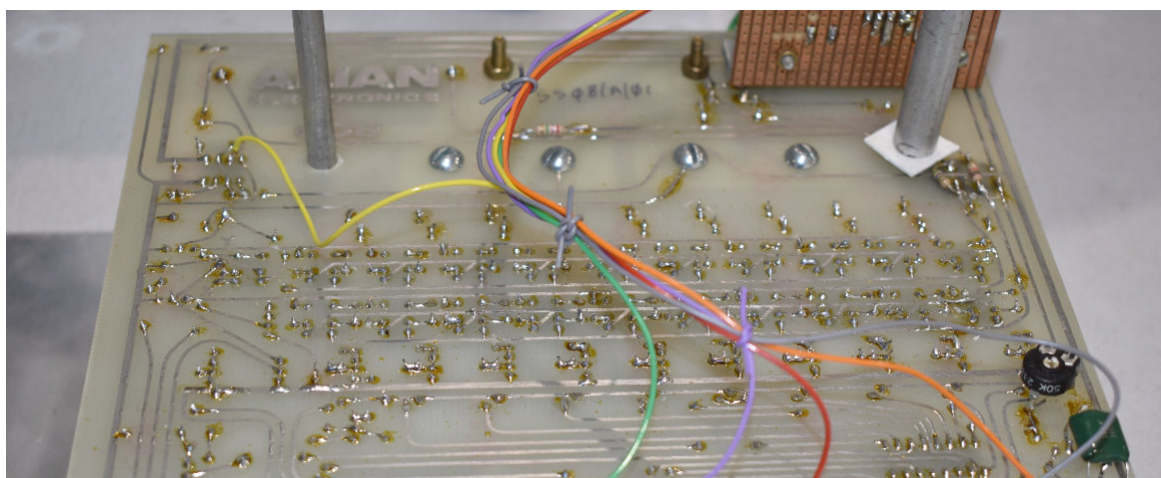
Not all radio amateurs want or need to build a printed circuit board. Many who follow the hobby for some time will encounter a reason to give it a try. The task has a pretty steep learning curve.

My first efforts using Computer Aided Design (CAD) was with Protel Easytrax in the early 80's. It represented a groundbreaking shift in the industry. Before CAD packages entered the market, circuit board design involved a ritual of putting thin black sticky tape and dots on panels of glass and applying a careful photographic process. It was slow and problematic. Design alterations were painful. It was with some relief earlier techniques were cast aside and the old 286 AT computer became a valuable tool. Protel Easytrax went through upgrades and takeovers and became what is known today as 'Altium'. This article doesn't favour any particular software package. There are plenty out there. Some are good and some are expensive and a few of them are both.

In this article we will look at what is really important and what is often touted as important, but isn't. It's a reflection of my own experience. Others may have a different take, but that's ok. I have around five hundred different pcb designs in the field and this body of work has produced some ideas worth sharing. Whether you are planning to make one or one hundred boards, the design steps are mostly the same.

Etch your own PCBs?

My advice is "don't do it". Your design time is valuable. Sure you can coat blank copper with photosensitive spray, prepare photographic negatives, assemble a UV Light box, develop the board and put it into a bubble tank of ferric chloride or ammonium persulphate until the excess copper is gone. Then break out the 1mm drill bit and make a few hundred holes. Been, there, done that and crossed it off my list. As can be seen in the cable test unit below, one of my early non-CAD efforts, the results will be ordinary. It won't have protective green solder mask, it probably won't be double-sided, won't have a printed component legend and won't have plated-through holes. Today, the tooling and chemicals you need to do this are going to cost more than the price of many manufactured boards.



Hand made circuit boards are not a good use of your time

Focus on your design, get your CAD file as good as you can and email your files to a specialist pcb maker. Two weeks later you will have some nice pcb's arriving in your letter box, far better than anything you can produce at home. (More about that later)

Numbers and Versions

New projects start here. If you design one board, there will be more. Sometimes they will be updates of the board you just made. Often they will be completely new projects. You really have to keep track of these somehow. Not just for yourself. Project titles will help your circuit board manufacturer to track your jobs and to replicate the correct version for future repeat orders. My recommendation? Pick a couple of letters. Perhaps name initials or a callsign suffix. It doesn't matter. Let's call it CAT. Your first circuit board project will be **CAT001v1**. If you make an update to change a part or add a feature it would be **CAT001v2**. If it is a new design it would be **CAT002v1**. You will refer to your board by this name when you save it to your hard drive and when you describe it on a purchase order to a circuit board manufacturer.

I began in the early 80's with pcb project AE001v1. In Sept 2024 this has reached AE485. (I skipped AE35 as this was the module in the 2001 movie that got the crew killed by the HAL9000. A bad omen.)

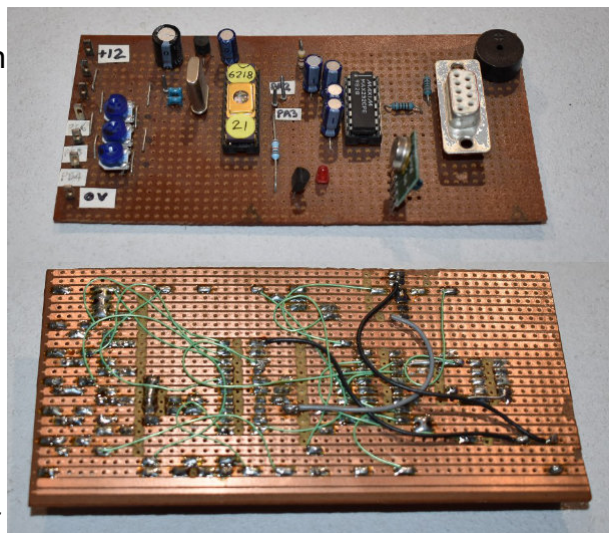
Don't add project descriptions in your project filenames. '**My_awesome_tesla_coil_version_1.pcb**' is a poor project management title. The pcb makers don't care what you are making and such filenames won't sort sequentially in your PC. Make an index sheet. Start a word docx table or spreadsheet and that's where you can put a longer explanation against your project code.

Don't bother veroboarding a full prototype of your project.

Many project builders will try to create their entire project as a rough prototype on veroboard, or other copper strip panels, before going to pcb artwork. This is very slow, can introduce many errors on a layout that is going to be discarded anyway. It is no fun trying to debug a rough version where components and values are not labelled. Certainly, if you want to advance-check a small aspect such as ideal op-amp or voltage regulator values on a breadboard, do that. Once proven and the circuit has been confirmed, put it aside.

Document your schematic circuit on paper or on screen as good as you can, then go straight to a version 1 circuit board layout.

When you get the board back and build up a proper Version 1, it will be much easier to test and trial this than wrestling with a lash-up job. If the project has software in it, you will want to know how it behaves on a board much closer to that which you intend to use. Ultimately, it is faster and cheaper than struggling to commission a half-assed version of the design. This leads us to the next topic:



Veroboard – you don't have to do this

Version 1 of any board will probably have issues.

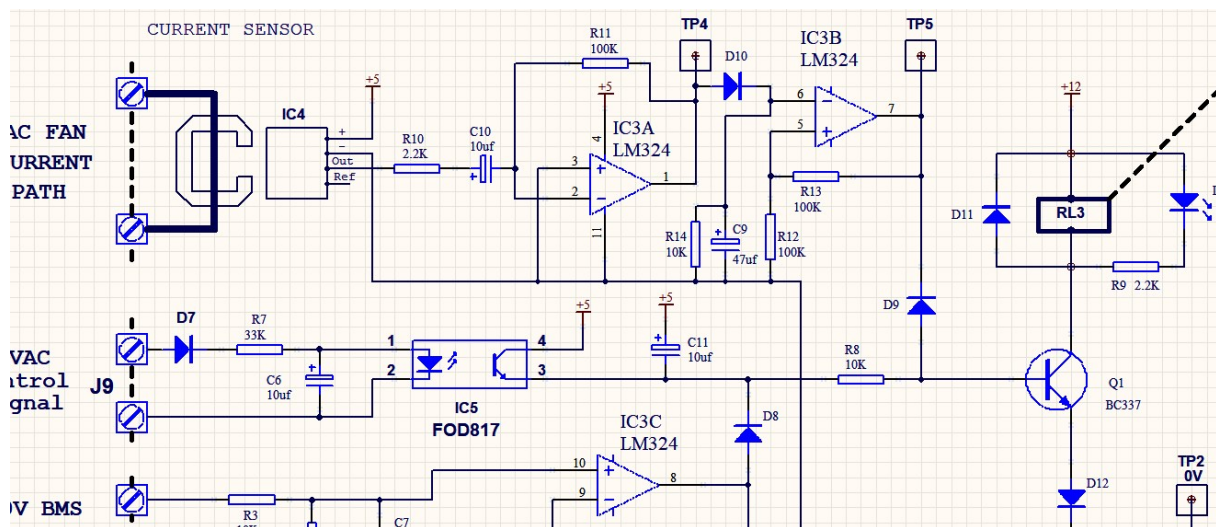
When preparing circuit board artwork it is ok to get your layout as good as you can before sending it off for production. However, when you get it back and put it together for the first time, you will usually find some unexpected surprises. Maybe the parts that you thought would fit, don't fit, or they conceal legend text. Mounting wires could be thicker than the holes size you had chosen. Perhaps there were alignment or physical collisions in the enclosure you selected. The point is that as much as you would like the board to be perfect, there will be changes. Accept this and learn from it. Immediately save your layout as v2 and make edits to it as you go, even if you never get proceed to a v2 in production.

Put a date on your Circuit board?

The circuit board should have the project code and version somewhere on the component overlay. Where sufficient room exists, adding the Year and Month of the design can be helpful. This is not essential, but it does tie the new design to a creation date. Handy in years to come.

Schematic diagrams

It is good practise to start the layout with a quality schematic drawing. Perhaps you have made a hand-drawn diagram and are keen to get into the board layout. Take the time to draw a proper schematic on an A4 sized page using the PCB schematic package. Ensure that every component has a part number (R1, R2, C1 etc.) You will have to make a good diagram anyway, so do it first.



Netlists

A Netlist is a list of connections that can be created by your schematic editor, detailing as a list of every component that is electrically connected together to other components or terminals. The theory is that a Netlist can later be reconciled against your circuit board layout and the program can alert you if some connections were left off the board. The principle is good and some designers swear by it. Others swear at it. My personal choice is not to use net lists as there can be a lot of work to get it



right, particularly where your design has a lot of special parts in it. I print a fresh copy of the schematic and as a track is placed on the board layout, I highlight that connection with a fluoro marking pen on the schematic diagram. Then keep going until every line on the diagram has been coloured in. It is a personal choice.

Set your board size

This is a bit of an art. Place most of your parts on a bit of paper, Push them around until the parts that will be electrically near each other on the circuit are close on the paper, with enough room for a few tracks to run between them. Where possible, orient all the IC's so that Pin 1 is towards the top of the board.

If size is not critical, be generous with the parts spacing. The art comes into it when you decide what sized enclosure you are going to put the board into. Make the enclosure no bigger than necessary, but don't go for too compact if you don't have to. Small boards are harder to design than big boards.

Selecting the enclosure first is really important. Good enclosures have circuit board mounts of defined spacing. Make your circuit board outline fit the enclosure, not the other way around, or you will finish up with a board that does not quite fit in many boxes, but rattles around in one that is much bigger than it ought to be.

Grid selection – Metric vs Imperial

Many beginners get this wrong. Most of the components are still manufactured on an imperial grid. Dual-in-line IC's and connector headers are usually based upon 0.1 inch spacing. Pads between pigtailed ¼ watt resistors are normally 0.4 inches apart. Capacitors are 0.1 or 0.2 inch pitch. However, larger parts and the mounting screws on enclosures will all be in Metric. So here's the trick. Start with your CAD package set to a Metric grid. Probably 0.5mm per step. Draw a line using one of the mechanical layers (not a track but a line) to outline the perimeter of the board. Still on the metric grid, Place large pads where the screws are going to mount at the four (or 6) mounting points of the enclosure. Once this is done, set your cursor to the Bottom Left hand corner of the board and define this corner as your **Point of Origin**.

Don't place ANY components on the layout yet.

Now it is safe to switch your CAD package to an imperial grid and set the step rate to 25mil. One mil is one-thousandth of an inch. With 25 mil steps, each pin of an IC will be exactly four steps apart on your layout. Use the Imperial grid for the rest of your design. (Later, when tidying up, it is ok to set the step to 5 mil when it comes to aesthetically moving component text to exactly where you want it to be.)

While safely on the 25 mil imperial grid, place all of your components on the board and roughly move them around to where you think they ought to be. Later they may have to be adjusted as the board layout progresses, but it is a good start.
(continued next page)

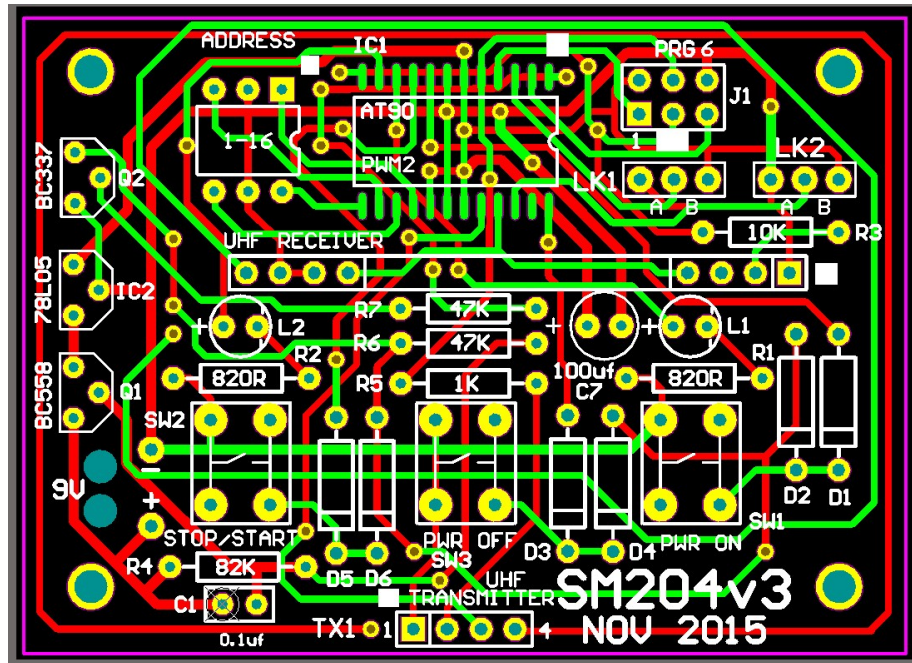


**BEING HEARD
IS IMPORTANT**

The Radio Amateur Society of Australia

Putting your layout together

In the example below there is a simple design for a hand held remote control on 433.920 MHz. It is used for fox hunts and clock adjustments, but that's not important right now. The artwork is here to serve as an example for several design aspects.



Component positioning & Text

All resistors, inductors and IC's have their value written under the part to assist with placement during construction and the part designation next to the component. They are well spaced. It helps if all diodes can be oriented the same way. Again for ease of placement and it sticks out if one was put in backwards when troubleshooting. If you don't have the footprint of the part you need, create one and save it to your library of favourite parts. How this is done varies somewhat between CAD applications. Don't trust parts libraries until you have physically compared the virtual part with the real part. Lots of component libraries have hole size and clearance errors. A digital vernier is an essential tool during this time.

Remember if you are doing a board with a component legend or writing on both sides of a board, that the text on the lower side is always mirrored during your layout. If both sides are readable during your design phase, then one side is going to look pretty stupid when you get your pcb back from the maker.

Note that a small square white square has been manually placed near Pin 1 of all IC's. This is optional but good practice so as to leave no doubt which way the chip goes on the board. The part outline shows this orientation, but it can also be obscured by IC sockets.

Think about what will make the board tough in the field. This board was powered by a 9V battery. In the bottom left there are two pads labelled + and – used for the battery clip on two wires. These wires will break off with very little flexing, so two holes have been placed next to the pads for the wires to thread through for strain relief. Larger cables directly connecting to a board should have two 4mm holes for supporting a small cable tie. It is attention to these little things which improves the reliability of the final product.

Track widths

There is a temptation to make tracks very thin, 10 mil or less, because they look good on the screen during design time. Get into the habit of making tracks as thick as the board will easily support. It makes the design tougher. In this example, which had a surface mount microprocessor, most tracks were 20 mil wide. Generally I make power rails (0V, +5V, +12V) wider at 50 or 100 mil, depending upon current demands. If I had to run a track between legs of a DIP ic, I would make the adjacent pads oval shaped and reduce that part of the track to 15 mil. Sure you can design your board with 5 mil tracks and get away with it. The pcb company will make it for you and maybe it would be ok. Unless there is no alternative, don't do it. Very thin tracks will make your final design unnecessarily fragile.

Component pad & hole sizes

Pads are what anchor your part to the board. Mostly resistors and small capacitor pads should be 65 or 70 mil in diameter, with a 32 mil hole. Diodes often have thicker leads and need bigger holes, so diode footprints have at least 70 mil pads and 40 mil holes (40 mil is about 1 mm). Headers for IDC connectors and programming ports also need 70 mil pads and 40 mil holes. I prefer DIP IC footprints to have round pads, leaving more space for tracks under the IC's. If you want a clean screw hole of say 3.5mm for a 3mm screw to pass through, it is ok to place a pad on the artwork and set the hole size to be larger than the pad.

Go easy on the cornering...

Don't put down tracks that turn corners at 90 degrees. It looks weird and is not the most compact way for laying down tracks. Take another look at the above sample board and you will see that all tracks turn in 45 degree steps. Much better. It improves track density. On double sided boards, ensure that tracks on the top layer don't go too close to holes for mounting screws. You don't want screw heads coming down on tracks where they can cut tracks or short them to a chassis.

Single or Double sided PCB's?

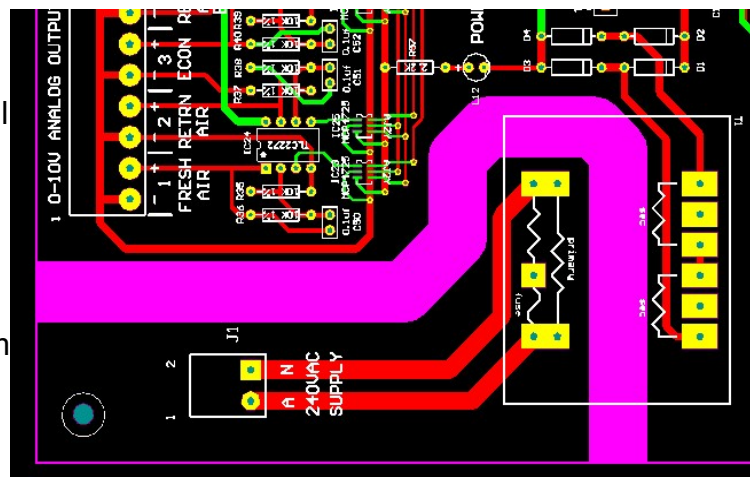
In the dark, early days, single sided boards with copper only on the underside of a layout were much cheaper than double sided boards. Today the price difference is not great, as the majority of circuit boards are now double sided. Plan on making all of your designs to be Double Sided boards.

There is another reason for Double sided boards. On Single Sided pcb's, parts pass through the board and solder to pads on the underside. The only thing keeping the pad in place is the glue between the copper and the fibreglass. Any vibration on the board, or brief impact to say a capacitor, can cause the pad to lift and shear away from the track that feeds it. Suddenly your project is intermittent and unstable.

Where the board is **Double Sided**, the interior of the hole is lined with a copper sleeve. That is your **Plated Through Hole** or '**PTH**' in industry jargon. When PTH pads are soldered, the solder soaks through the hole and anchors to the pad on the top side of the board. This is much stronger. Double Sided boards are always going to be significantly tougher than single sided boards. I made plenty of modules for early golf cart electronics and saw what came back over time. (particularly if they hit their electronics with a club after a bad shot) Board integrity failures fell to zero after a transition to double sided boards was made. Today, even if I were making a design with no tracks on the top of the board, I would still order it as a double sided board with plated through holes. The small price difference for the avoidance of future failure is money well spent.

High Voltage vs Low Voltage areas

Perhaps your design has a small power transformer on it with a 240V primary, or a relay which will switch 240V. You will want to keep all mains powered tracks at least 10mm away from your low voltage vcircuitry. I like to place a wide track or fill 10mm wide and put it on the layer which is a **Solder Mask Void** layer. This path will go between the high and low voltage zones on the board and make it clear to anyone working with the module that compliant isolation is present.



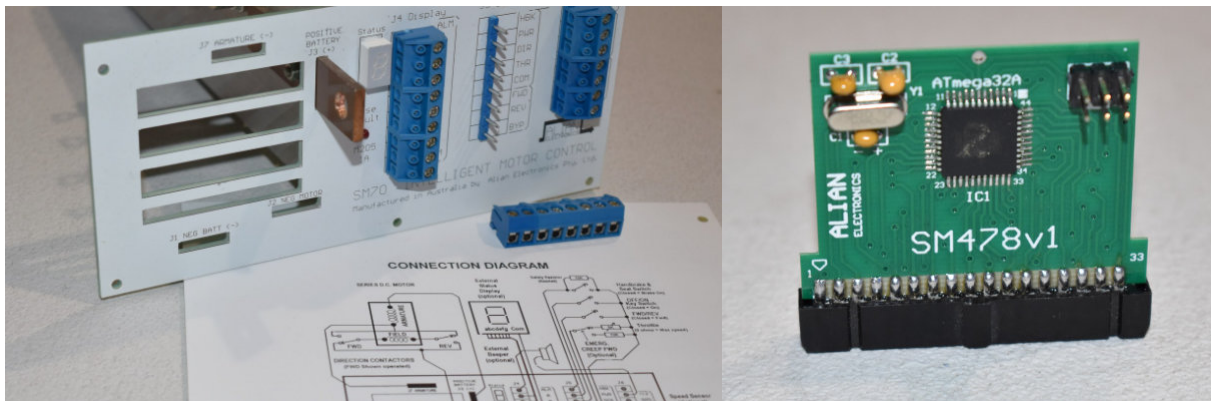
A clearly defined 10mm high voltage safety barrier in the lower solder mask

Auto-Route?

Auto-routing of board layouts have been around for a long time and it has gotten pretty good. With this technology, you put down your parts, highlight pads to be connected and let the program draw your tracks for you. Try it if you like. Personally, I don't use this feature. I prefer to manually insert my tracks. It forces me to think harder about component placement. For auto-routing to work properly, design rules and net lists have to be spot on. This also takes time. After auto-routing, some corrections and human interventions is still going to be necessary in complex areas of a layout. (Like cars, driving an auto is fine, but if you really want to understand what your car is doing, drive Manual first.)

PCB board & copper thickness

The standard thickness for a fibreglass circuit board is 1.6 mm thick and the copper is 35 microns thick. This is fine for 95% of jobs. There have been times when I have ordered boards to be 1mm thick, so that it can fit on to IDC header plugs. (shown below) There have been projects where the circuit board had been part of the structure where it was ordered as a tough 3mm thick board, as with this forklift motor controller. We can afford to be creative with this work. The forklift unit had a 3mm pcb lid with zero trackwork, holes pre-drilled, white solder mask and customer wiring diagram printed on top.



(Left) a 3mm thick PCB as a lid and a front panel

(right) A 1mm thick pcb as an unpluggable micro

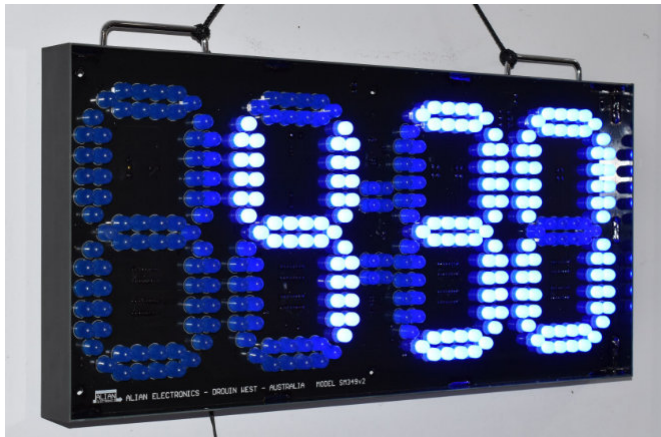
There will be some occasions where high current flow is a factor. In some golf cart pcb's, wide tracks on top and bottom of a board are run in parallel to double the current capacity. Optionally, the pcb can be ordered with copper 70 microns thick instead of the usual 35, which will double the current capacity again. These are all possibilities.

Multi layer boards?

Most big pcb companies offer multi-layer boards where there are extra copper layers between top and bottom layers. This can increase the component density of a board, or offer a shielding layer to RF sensitive boards. I do not usually design such boards. They do get expensive. Should something happen to a track in a buried layer, the board cannot be serviced. By all means try it, but stick to double sided boards wherever you can.

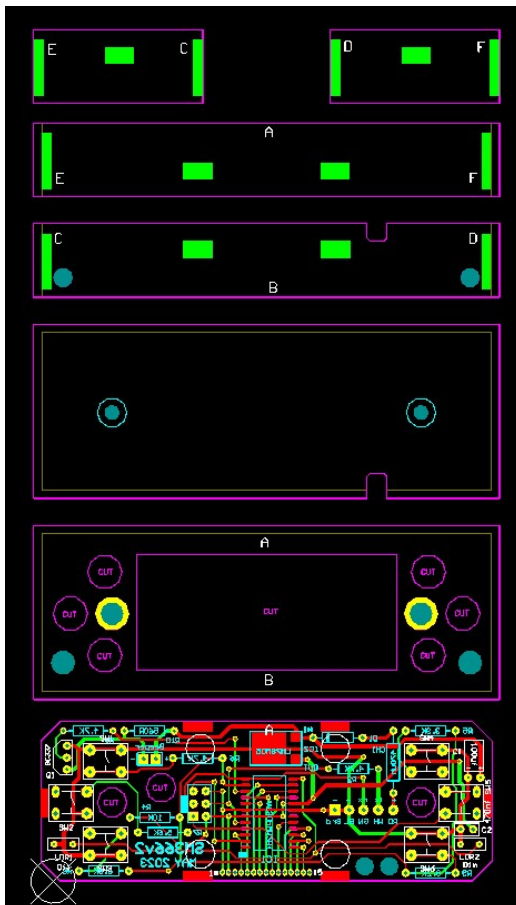
Solder mask & component legend colours

When ordering your circuit boards you will be asked to nominate a solder mask colour and a component legend colour. The default colours are pretty much white text on a green protective solder mask. That's fine. Mostly people won't see it anyway. Sometimes it's nice to do something special and go for white writing on a black solder mask, like this blue LED clock assembled for the next Antennapalooza event. The side panels of the clock which hold the acrylic screen are also pcb material. In the right situation black text on a white solder mask can be very effective.



A Black solder mask with white writing can make a project look good

PCBs as building materials

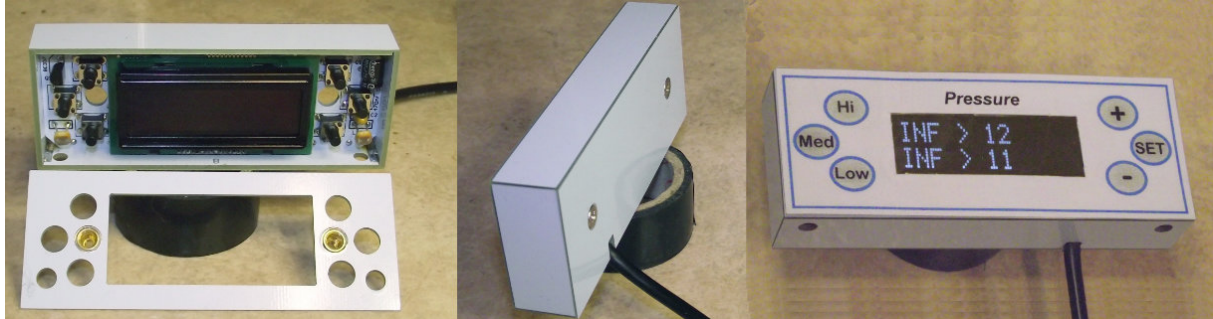


As seen in the previous section, it is quite feasible to use circuit board as an enclosure construction material. You get something that is tougher and more UV stable than 3-D printed plastics. Certainly it is a lot cheaper than getting dies made for injection moulded plastics. In this example the pcb layout can include multiple panels of the enclosure, with all holes and cutouts prepared in the layout, plus the circuit board itself that sits in the centre of the enclosure.

There are copper fill areas on the ends of each panel allowing the enclosure to be assembled by solder deposits on the inside. The edges of the front and rear panels have had the first 1.6 mm routed down 1, deep, so that when the enclosure is screwed together, the corners of the box come together neatly.

(Left) You can design & order your PCB as a set which includes the enclosure

The front and rear panels clamshell together with 3mm screws. Push button switches come up flush with the front surface so that the label will flex when a button is pressed. The two holes in the base of the enclosure were for photocells to dim the OLED display at night. Sure you have to put it together, but this is still faster, neater and more repeatable than drilling a bunch of holes yourself in a plastic box.



If the edges of front and rear panels are machined, you get a neat box

Pre-order integrity checks

If you have completed the pcb layout and you think it's about right, then there's another step that may save your project from tragedy. On the screen, turn off the component legend layer and the top copper layer. Now you should see the bottom layer tracks and pads in the clear. Carefully inspect the board. Are clearances to pads and other tracks good? Could some tracks be centred a little better? Did you lay a track around something that is no longer there? Maybe it's ok to make a track take a more direct path on this layer. Did a track pass through a hole or cutout? Such issues are reasonably easy to spot with this view.

Next turn off the bottom layer, turn on the Top copper layer and repeat the exercise. When that is done, turn both copper layers off and activate the component legend to see how the text is going to look on the board. For this layer, I set the layout grid to just 5 mil, so that I can more accurately position the text next to the parts. Ensure that the text is not on top of any vias or pads, as that text will be lost. Ensure that all the screw terminals are correctly labelled. Add any other relevant text if there is room, such as what option links do when they are in or out. When all this is done you are ready to send your file to a pcb maker.

Pick a pcb company

There used to be many circuit board manufacturing companies in Australia. The quality was good, but they were slow. One by one they disappeared. Tooling and production costs were very high. I patronised Australian manufacturers for as long as I could, then discovered that some companies were just taking my orders, quietly getting the boards made in China anyway and charging an extra 100% for the handling.

For the past 20 years my boards have come from companies based in Shenzhen China. There are many of them. Mostly I use WHX Circuits. The customer experience has been good. (In 2017 I went there in person and they gave me a factory tour and a free lunch. I enjoyed the tour, but the deep fried chicken feet, duck tongues and warm beer, not so much.)

When ready, I email a pcb file, along with my requirements (See pcb ordering notes) . Within about 2-3 hours, I have an invoice to pay. Within another couple of hours they do a technical assessment and ask questions if there are anomalies spotted by the production team. A track that goes nowhere? (Yes that's supposed to be an antenna track) Should these holes be plated through? Sure why not. The written email English by the staff in these companies is quite good and they have been universally supportive.

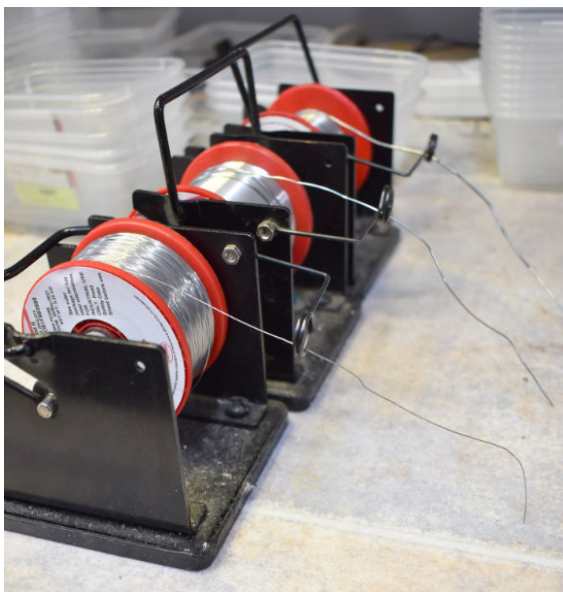
China makes excellent circuit boards for a great price



Within 5 days the job will be done, whether it is 3 or 300 circuit boards. There will be a Tooling fee, a price-per-board fee and a delivery fee. Delivery time is typically 5-10 days later. Be aware, if your order is near Chinese New Year or other national holidays, there will be extra delays.

Pricing will vary, but typical examples in 2024 are \$50 AUD for tooling fees and \$3-10 per board depending upon size, hole quantity and some other options. Usually the best way to pay is Pay Pal or credit card. As a company, paying overseas suppliers by Transfermate works out cheaper than most other methods.

Solder size and type (PB or not PB?)



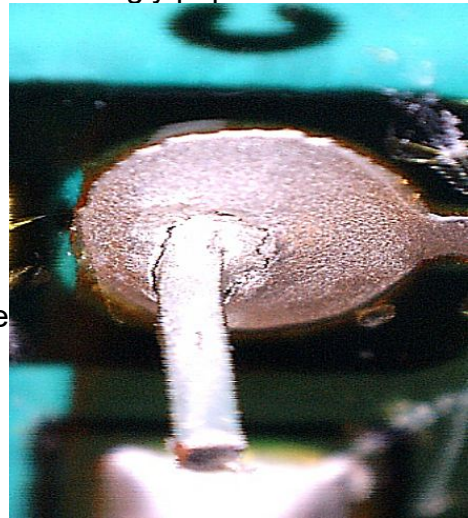
3 different solder sizes for 3 different component types

When assembling a circuit board, we need resin cored solder. It is important to have different sizes on hand. 1.2mm solder is appropriate for all of the big parts, fills, connectors and flying wires. 0.7mm solder is used for 90% of most boards. Great for IC's and regular discrete components. I also like to have a reel of 0.3mm solder for the surface mount parts that have become a major part of the electronics industry.

Having a nice temperature controlled iron is very useful, however the right size solder is critical. If I had a choice between a good pointy iron with oversized solder, or a big ugly iron and the fine solder, I would go with the fine solder every time. (As it says on the T-shirts, if it smells like chicken, you're holding it wrong)

We have a choice between using Lead Free solder (96% Silver) or regular solder (60% tin, 40% Lead). Lead free solder has become increasingly popular and is now mandatory in Europe. Often it is called **PB Free** solder. **PB** is the element symbol for Lead. It is an abbreviation for **Plumbum** which is apparently Latin for lead. (Early Roman builders used Lead for **Plum Bobs**. A weight on a string. Another thing that the Romans did for us...)

The problem with Lead free solder is that it has a higher melting point. Irons have to run hotter. The hot flux can spatter on your hands and the joints have a matt, slightly crystalline appearance. Cracks and dry solder joints are a problem, as can be seen in the adjacent microscope image. My preference for projects and small production work is to use the original 60/40 solder, which cools with a nice shiny finish. Also, soldering iron tips will last longer with lead based solder.



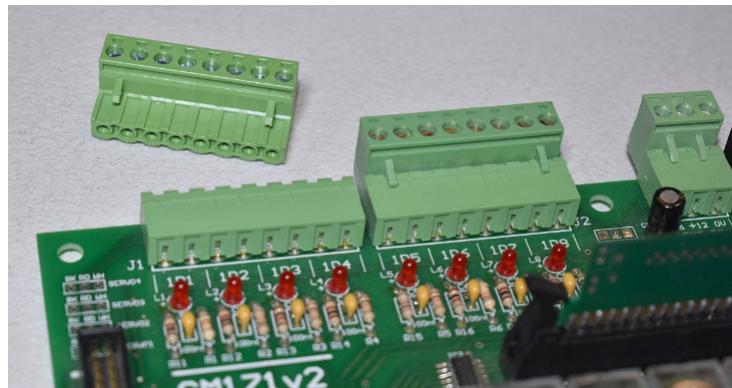
This bad Lead Free soldering has microscopic cracks

IC Sockets

Contrary to most industry convention, I tend to use IC sockets for all DIP IC's. There is a cost per socket. Perhaps 20 cents, but you only have to want to change one chip once and the 20 cent socket cost has become a sound investment. (It takes time to remove and re-solder entire IC's just on spec that it could be a crook device.) If you decide to make a Version 2 of a circuit board, it's easy to remove and re-use all the socketed chips from your obsolete version 1 board.

Terminal blocks and Wires

Your design will have wires going to or from the board. I don't like wires that solder directly to circuit boards. Screw terminals that can be unplugged are best. If the board must be removed, swapped or placed in a test harness, it can be unplugged without losing the positioning of the connecting wires, preventing future faults. Where there is more than one such connector, ensure that they are of different sizes or number of terminals, so they can't be swapped accidentally. Leave room for identifying text on the function of each terminal, so it can still be easily read when the connector is plugged in.



Screw terminal blocks are much nicer than soldered wires

Filling areas with copper

Many designers like to use flood-fill tools to extend a ground plane track work to all parts of the board not occupied by tracks and components. For radio applications, the extra grounding can improve stability. For the rest of the time, the extra ground



plane is a distraction and increases the chances of tracks shorting to earth. I don't often use this feature.

Completing power rail loops

If you have a negative rail, or +5 or +12V power rail traversing your board, try to loop the track ends so that they meet and form a complete circle. It lowers the impedance of the supply and reduces voltage drops on supply rail tracks when they are under load. It's not always possible, but it can improve stability if you have the track room to do it.

Getting boards cut out, panelised, snap-out or V-grooved

The circuit board maker may ask if you want boards to be joined together in a larger pcb frame that you snap out when ready. The snap areas are either a series of fine holes, or a section that has been partially cut through in the shape of a 'vee'. Mostly this is harmless and makes them look like they came out of a 1960's cereal packet. Generally I would request that the boards are fully cut from their supporting frame and are sent ready-to-use, without having to snap them from a frame.

Placing an order

With any new order to a pcb manufacturer, create a one-page Word document that will accompany the artwork file. It should contain the following information:

- PCB filename/version CAT003v2
- An order reference number 1234
- Your contact details, email and delivery address.
- New order or Repeat order (If 'Repeat', you don't need to supply the artwork file)
- The name & version of the pcb package you used to create your layout
- How many boards you want
- Height & width of the board in mm
- PCB thickness Usually 1.6 mm
- Copper thickness Usually 35 um
- Hole quantity Approximation is ok
- Smallest hole size Usually 28 mil in a via hole.
- Thinnest track Check your board. 20 mil is good.
- Layer format Single/Double sided, through hole plated
- Slots or large holes Yes/No

- Layers to ignore

Some mech layers may have been added to there to help you position enclosures and external hardware,
The pcb maker should be told to ignore them.

- Solder mask colour

Usually green

- Component legend colour

Usually white

- Comments

Any notes that may help the manufacturer.

Clear laquer?

About half the boards I put together will find themselves in a humid, harsh environment, where dirt can get in, ants & spiders nest, or some idiot wants to wash the pcb with a high pressure hose. In those instances, I cover the connectors (and IC sockets if not populated) with masking tape and hit both sides with a clear pressure pack lacquer. The Chemtools ACC400 from Radio Parts is really good for this. Fast drying, clear and easy to solder through if you must do a repair. It does not make a board waterproof, but it does make them last a lot longer in the field.

One extra hole...

Although not part of a circuit board design, consider putting a couple of tiny holes in the bottom of your waterproof enclosures. Remember, nothing holds water better than a waterproof enclosure. I used to make 1mm holes, then I encountered a race of 1mm ants. So now I drill 0.7mm holes in boxes.



You want ants? That's how you get ants

Summing up

This article is already a long one, but contains a lot of information. A topic this big could easily have become a book, but I lack the time to write it and you probably lack the time to read it. It is intended for the experimenter who is prepared to go to the next level. Perhaps it will help with a single project or maybe it will launch a new career. That's up to the reader. Like any skill of this nature, the more boards you design, the better they become. Give it a try. You won't get bored.



***The Radio Amateur
Society of Australia***